International Journal of Pressure Vessels and Piping 117-118 (2014) 9–16

Contents lists available at ScienceDirect

International Journal of Pressure Vessels and Piping

journal homepage: www.elsevier.com/locate/ijpvp

The effect of analysis model on the stress intensity calculation for the nozzle attached to pressure vessel under internal pressure loading

Ming-Hsien Lu*, Jiun-Shya Yu, Jien-Jong Chen

Institute of Nuclear Energy Research, Atomic Energy Council, Taiwan, ROC

Keywords:
Nozzle
2-D axi-symmetric Model
Vessel radius

ABSTRACT

The stress behavior of a nozzle attached to the pressure vessel is usually analyzed using a simplified 2-D axi-symmetric model, where part of the vessel is also included in the numerical model. For conservatism, the modeled vessel radius sometimes is larger than the actual value. This paper creates three different 2-D axi-symmetric finite element models, where different vessel radii are modeled, i.e. 1, 1.5 and 2 times the actual vessel radius. Using these simplified numerical models to calculate the membrane and membrane plus bending stress intensities along some selected sections when undergoing internal pressure loading, and comparing these results with those evaluated from the realistic 3-D model, it shows that the 2-D model with vessel radius equaling to the actual value could well represent the behavior of a nozzle attached to the vessel.

1. Introduction

The reactor pressure vessel is made from cylindrical shells with two hemispherical heads on the top and bottom sides. Bottom head is welded to the lower cylindrical shells, while top head is bolted to the flange of upper cylindrical shells. Nozzles like Feedwater nozzles, core spray nozzles, or recirculation nozzles are normally intersected with the reactor vessel cylindrical shells. The whole structure is not only the pressure boundary but also the first barrier to confine the radioactive materials. Its structural integrity is critical to the safety operation of the nuclear power plant, therefore ASME code classifies the reactor vessel and attached nozzles as class 1 components, and requires certified detail stress reports to prove the safety operation of the reactor vessel and attached nozzles during the design life.

ASME Code Section III [1] specifies the stress limits on membrane and bending stresses for class 1 components. Usually the membrane and bending stresses along some important sections (or lines) selected from the numerical model are calculated and compared to the Code requirements. Hechmer and Hollinger [2–4] used 3-D finite element models to study the stress behavior of a nozzle-to-cylinder intersection structure, and suggested the sections to be selected. In their recommendation, the selected sections should not locate in the “ring” juncture. However for the design/analysis of the class 1 components in nuclear power plants, the 2-D axi-symmetric models are commonly adopted and the stresses along the nozzle to vessel juncture are usually evaluated since the significant stress exists in this area.

Fig. 1 shows a 3-D solid model consisting of a nozzle attached to the pressure vessel. When using a 2-D axi-symmetric model to simulate the nozzle, part of the vessel structure is also included in the model to consider the vessel effect on the stress behavior of the nozzle. However the real stereoscopic structure of this axi-symmetric model is a nozzle attached to a spherical shell structure (Fig. 3) rather than attached to a cylindrical shell structure. The calculated stress results of this simplified 2-D model could not exactly reflect the real situation, but give an approximate stress condition, especially in the nozzle to shell juncture area. To avoid under-estimating the stress values, analysts assumed different vessel radii in the axi-symmetric model.

This paper establishes three simplified 2-D models and denotes them as models A, B, and C of which the simulated vessel radii are 1, 1.5, and 2 times of the original vessel radius respectively. Applying the same internal pressure loading to those 2-D models and calculating the stress intensities for some selected sections, the analyzed stress results are compared to the target values evaluated from a real 3-D finite element model so that the appropriate vessel radius modeled in the 2-D axi-symmetric simulation could be determined (Fig. 2).

* Corresponding author.
E-mail address: mhlu@iner.gov.tw (M.-H. Lu).

0308-0161/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijpvp.2013.10.004

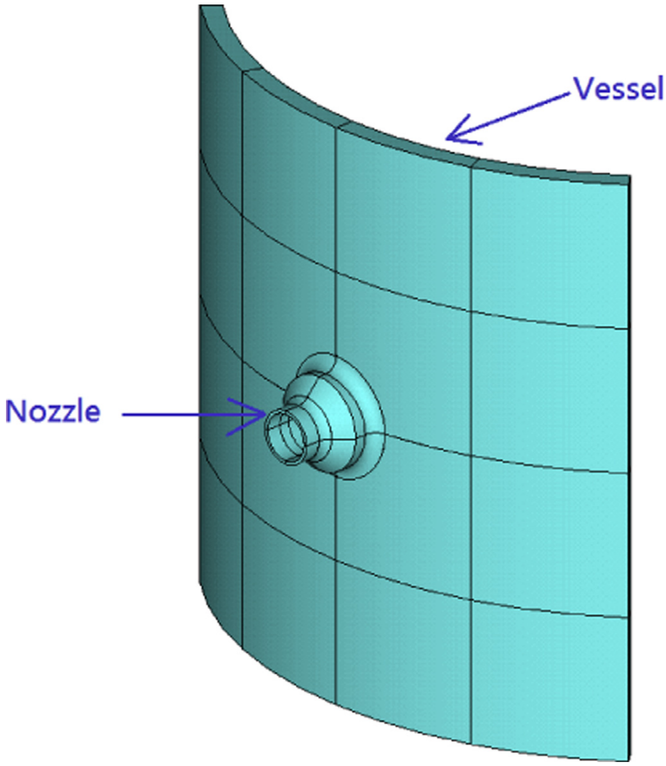

Fig. 1. The configuration of the reactor vessel and attached nozzle.

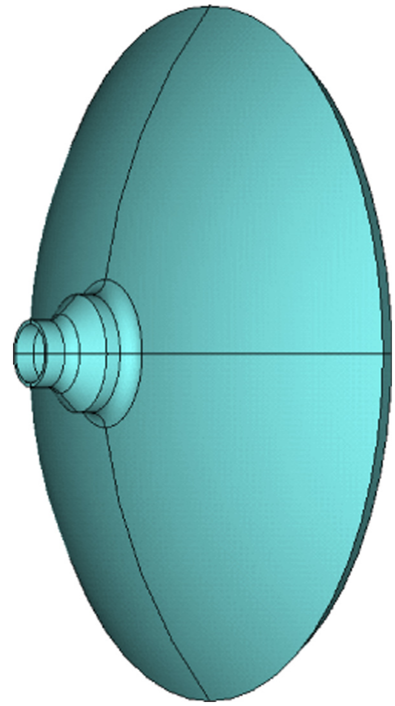

Fig. 3. 3-D stereoscopic model presented by 2-D axi-symmetric model.

2. Nozzle geometries and analysis models

2.1. Nozzle structure

This study assumes the inner and outer diameters of the vessel shell are 203.36 inch and 213.52 inch respectively. Fig. 4 gives the detail geometry sizes of the attached nozzle, where the inner and outer diameters of the nozzle are 11.66 in. and 13.27 in. respectively.

This study also assumes that both the nozzle and vessel shell are made of SA-508-2 steel material, and the corresponding elasticity modulus and Poisson ratio are 25.43×10^6 psi and 0.3 respectively.

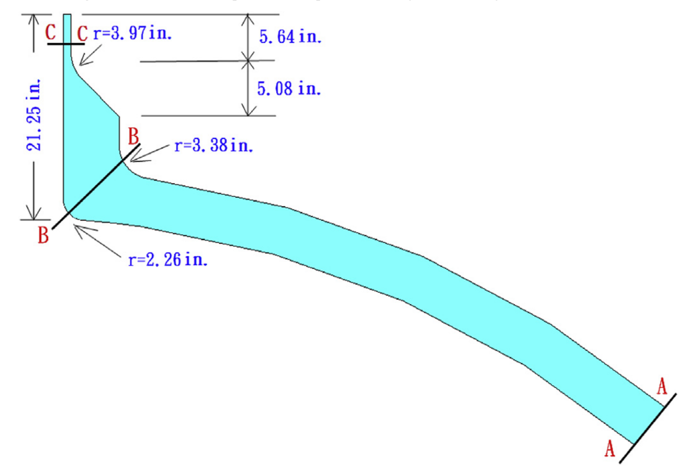

Fig. 4. The measurements of nozzle which welded to vessel shell.

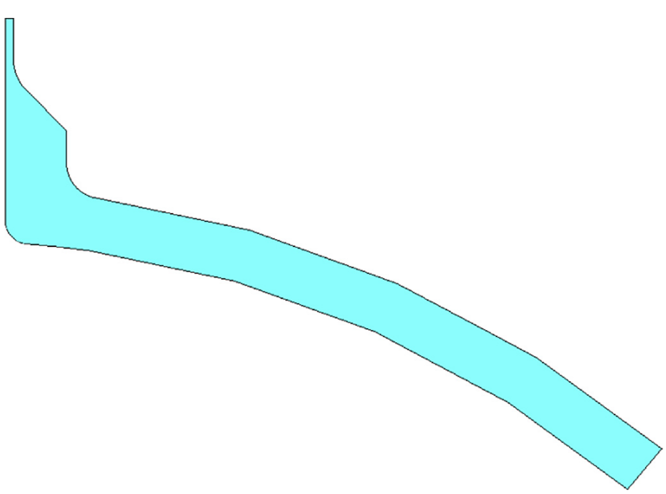

Fig. 2. 2-D axi-symmetric model.

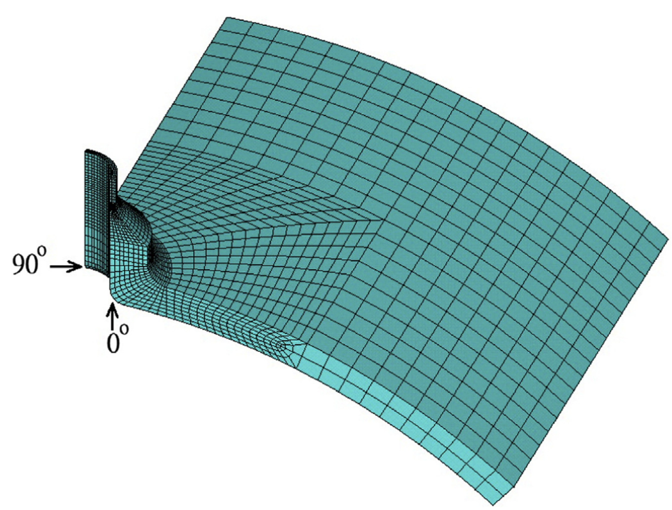

Fig. 5. 3-D Finite element model.